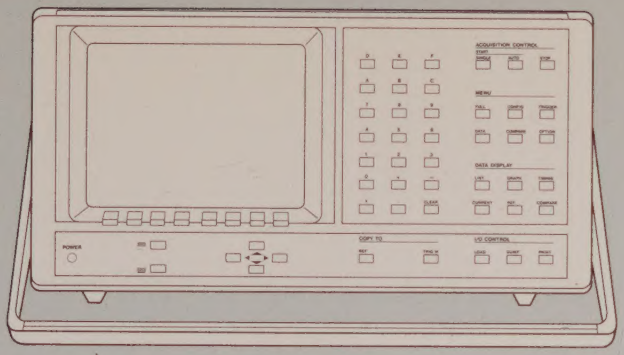


PHILIPS PM 3570

MP MASTER
MA24616

QUICK OPERATING GUIDE

PHILIPS
Test &
Measurement

PHILIPS

CONTENTS

I	**Introduction**	
	1) How to use this guide	1
	2) Start-up and Quick Run	2
	3) System configuration	5
	4) Hook-up to the target	6
	5) The configuration menu and the option menu	9
	6) Printing data	11
II	**State analysis**	
	1) Introduction	14
	2) Making and displaying an acquisition	16
	3) Displaying state data (disassembly listing)	18
	4) Selective data acquisition	20
	5) Comparison of state data	22
III	**Timing analysis**	
	1) Introduction	26
	2) Making and displaying an acquisition	26
	3) Timing triggering	28
	4) Comparison of timing data	30
IV	**Time correlated analysis**	
	1) Introduction	34
	2) Making and displaying an acquisition	36
V	**Performance analysis**	
	1) Introduction	42
	2) Event frequency analysis	44
	3) Execution period analysis	46
	4) Displaying results in a histogram	48
VI	**Appendix A**	
	Menu diagram	53

Digitized by the Internet Archive
in 2025

https://archive.org/details/manualsplus_10074

HOW TO USE THIS GUIDE

The frontpanel of your analyzer, can be divided into three main parts.
The first part consists of the monitor, the softkeys and the cursor control keys. The keys of this part are shown in yellow when used in this guide. The second part is the hexadecimal keypad, coloured blue, and the last part is composed of the "analyzer control keys", shown in red.

When using this Quick Operating guide, you may place it on top of your analyzer. Now you can select two ways of working: The first one at the left page with the text and explanations, or the second one at the right page giving you the keys used and the corresponding displays.

If necessary you can use the side flap, which shows a representation of the front panel with the keys coloured in the 'Quick Operating' colour.

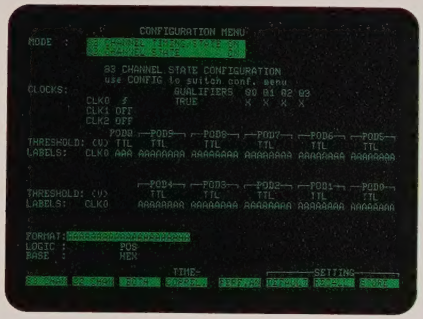
In this booklet an educational target based on an M68000 microprocessor and installation of M68000 software and hardware has been used. However, any other target system can also be used with this booklet, using its value as an Operating guide for proper operation of your analyzer.

Note: *In all the different chapters the default state of the PM 3570 logic analyzer is taken as starting point. If you are not sure, your instrument can be put into this default state by pressing the softkey DEFAULT in the configuration menu.*

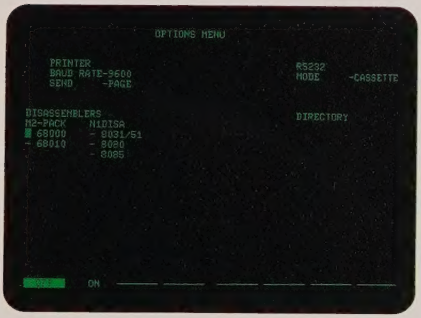
1) Switch your analyzer on with the switch on the rear panel.
2) Push DEFAULT — Soft key
3) Push OPTION Hard key
4) Place cursor on 68000 with control keys (or on other disassemblers when using another microprocessor)
5) Push ON
6) Push the CONFIG key and note that the configuration is set for your microprocessor. Now your analyzer is ready for measurement on your target. We will take this as the starting point in this booklet.
   Of course you can make a Quick Run by doing the following:
7) Press SINGLE and an acquisition is made
8) Press LIST to display a state list or press TIMING followed by BEST to display a timing diagram.
9) Press below screen until DISA soft key appears

DISPLAY ON SCREEN

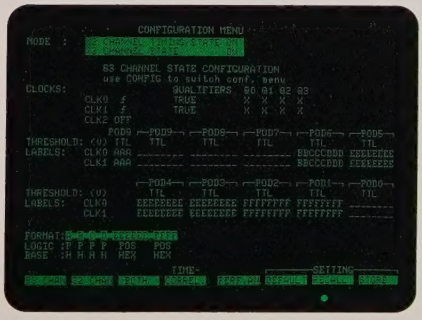
KEYS USED



1) Switch the analyzer ON
2) DEFAULT



3) OPTION
4) Move ◁▷ cursor to '68000'
5) ON



6) CONFIG

To make a quick run:

7) SINGLE
8) LIST or TIMING and BEST

SYSTEM CONFIGURATION

The setting up of the PM 3570 logic analyzer obviously depends on the facilities provided by your instrument. This Quick Operating guide describes some of the ways in which a PM 3570 with the following configuration can be used:

- 83 state channels
- 32 timing channels
- RS 232 C interface/printer interface
- several disassembler packages, including the M68000 disassembler

HOOK-UP TO THE TARGET

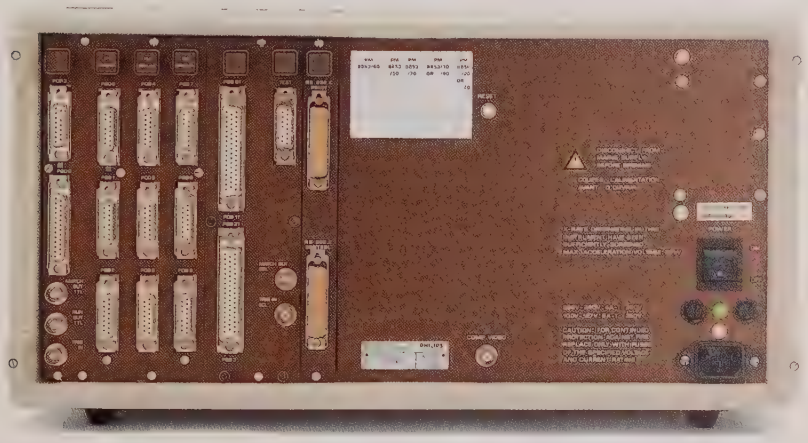
The intention of this chapter is to explain the installation of all the pods to the microprocessor of your target.

- First you should plug your clip onto the microprocessor, and then connect the personality adaptor to this clip.
- The personality adaptor provides several output connections.

These are connections used when connecting the different pods to the analyzer.
As described in the Operating Manual (chapter 8, part M2: disassembler for M68000 and M68010) we need the pods Q, 6, 5, 4, 3, 2 and 1. Pod Q is the clock pod, pods 1 and 2 provide the data lines of the microprocessor, pods 3, 4 and 5 provide its address lines, and pod 6 provide the status lines.

Then proceed as follows:

- Plug your 32-channel single probing adaptor (PM 8827) into the connectors of the timing analyzer module (PM 8853/30 or /90)
- Now plug its four connecting cables as follows:
  1) first cable from top into the connector of state pod 1 (on PM 8853/60)
  2) second cable from top into the connector of state pod 2 (on PM 8853/60)
  3) third cable from top into the connector of state pod 3 (on PM 8853/60)
  4) last cable from top into the connector of state pod 4 (on PM 8853/50)
- Plug pod Q into the connector of state pod Q (on PM 8853/60)
- Plug pod 6 into the connector of state pod 6 (on PM 8853/50)
- Plug pod 5 into the connector of state pod 5 (on PM 8853/50)
- Plug the first dual timing pod into the top connector of the probing adaptor. The pod above is your pod no. 1 and the one below is pod no. 2.
  The second dual timing pod provides you in the same way with pod no. 3 and pod no. 4 when plugged into the down-connector of your probing adaptor.



With this configuration, you are ready to measure in timing mode the data lines and the lowest address lines of your M68000 microprocessor. You might also want to observe other channels. If, for example, you want to see the channels of pod 5 and pod 6 in timing, instead of the channels of pod 2 and pod 3:
Connect the channels of pod 2 and pod 3 with state pods to the target and use the remaining timing pods for the channels of pod 5 and pod 6.

Configuration menu

By switching on the analyzer with the ON/OFF switch on the rear panel, the state configuration menu will be displayed on screen. This is the basic menu of the Logic Analyzer. You may also decide to do Performance Analysis at this place.

After pressing the button CONFIG again, the monitor displays the configuration menu used during the Timing Analysis.

The config key has a "toggle" function. If you press it you will see the state configuration menu and by repressing it the timing configuration menu. This key function is also used in the data menu and the trigger menu (by DATA and TRIGGER).

Option Menu

Before starting the analysis, you should initialize the analyzer for the microprocessor you are going to use. This can be done in the option menu, which is reached by pressing OPTION. Here your define the configuration needed for the microprocessor under test by first moving the cursor to the required disassembler and later pressing the softkey ON. Moreover, this menu allows you to select the baud rate for your printer, and you can set the option 'print hardcopy', which allows you to make a screen-dump at any time simply by pushing the PRINT key.

PRINTING DATA

For printing data an RS 232 C printer interface is available. This output can be found on either the PM 8856/20 or the PM 8856/40 module. These must have been installed in slot 7 at the rear of the analyzer.
The connector you need for printing data is the top one. For the functions of the bottom connector, see the operating manual chapters 7 and 9.

After connecting your printer, select the OPTION menu. Then select the necessary baud rate for your printer. Be sure that the baudrate setting of your printer and the logic analyzer interface are the same.

You can also select the kind of printer dump to perform:

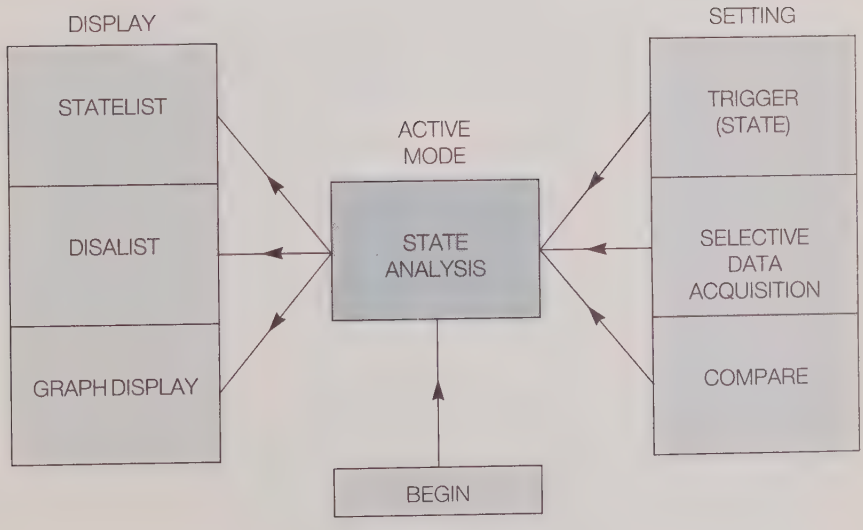
1) printing of the acquired data
2) printing of the menus used (settings)
3) making a screen hardcopy
4) printing just one page of information

Pushing the button PRINT starts the printer program, and by pushing it again stops it immediately.

Note: If you are not sure how to connect the logic analyzer and your printer, a description of this connection is given in the operating manual chapter 7A, part 3.

STATE ANALYSIS

1) Introduction
2) Making and displaying an acquisition
3) Displaying state data (disassembly listing)
4) Selective data acquisition
5) Comparison of state data



Introduction

While performing state analysis, the analyzer samples data into its memory under synchronous control of the relevant clock signals from the microprocessor.

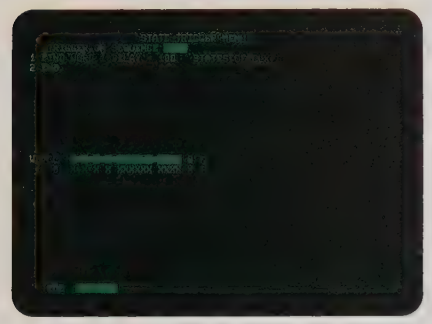
To demonstrate this on your analyzer:

It is assumed that you proceed next to the start-up procedure described.

1) First choose the state analysis mode by pressing BOTH in the configuration menu (you can use 83 CHAN. only if you do not use the PM 8827 single probing adaptor).
2) Press TRIGGER to enter the trigger menu. In this menu you can create your own state trigger sequence. On screen the default trigger menu is shown. When using this trigger sequence, the analyzer will act as follows:
   Proceed with acquisition until word 1 is found. As soon as word 1 appears, the analyzer continues acquisition for 100 states of clock 0. This means it fetches the incoming data untill the 100th sample of clock 0 appears: of course the analyzer also puts all the samples of clock 1 into its memory, but the end of the acquisition does not depend on these acquisitions.

DISPLAY ON SCREEN KEYS USED

2) [illegible]



2) TRIGGER

Making and displaying an acquisition

Now the system is ready for an acquisition. To start press

1) SINGLE. The analyzer receives data and stops at the moment it detects the trigger word. Now the data present in the memory can be listed on screen by
2) LIST.

The analyzer triggered at the address you have defined. Because the default is set to don't care, any value of the trigger word is sufficient, but you may wish to select another triggerword.

Changing the trigger word with COPY to trigger word

3) Press the ▤ -key
4) Scroll up or down with the cursor keys or use PAGE+ or PAGE– untill the required triggerword is found
5) Press the ▤ -key
6) Place cursor on the trigger word
7) Press COPY TO TRIGW-key

If for instance you are, only interested in the addresses and their data fields, you can blank the other state columns using BLANK.

First press

8) ▤ -key, enter format mode by pressing
9) FORMAT and afterwards remove the status lines with
10) BLANK.
    If you want to recall them, use RECALL.

You can also invert the value of a column by INVERT. The label will be changed to a lower-case character. By pressing INVERT again, the original column will appear. Finally, leave the format mode by pressing POSIT to return to normal mode.

DISPLAY ON SCREEN KEYS USED

1) SINGLE
2) LIST
3)
4) SCROLL WITH OR
OR to trigger word

5)
6) place cursor ON trigger word
7) COPY TO TRIGW

8)
9)
10) and RECALL THEM WITH
FINISH WITH

Displaying state data and disassembly listing

After making an acquisition, and having listed the results by LIST, you may be interested in the assembly code of the data received. This takes place by using 1) DISA.

Displaying the programflow

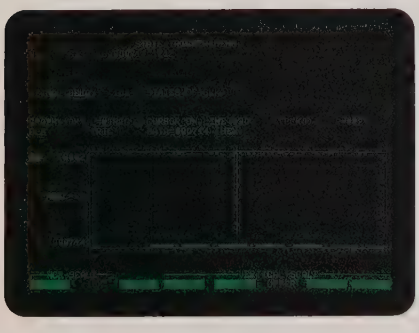
Also, if you wish to get an idea of the program flow, it is possible to show a graphic-display of the running program on screen.

Use:

2) GRAPH
3) 10X
4) LABEL and afterwards E because this is the address column.
5) BEST and follow the flow with your cursor.
6) You can alter the maximum address simply by using the softkeys; first UPPER (notice the arrow beside the maximum address), then press (7) CHANGE and (8) enter the required maximum address. Stop the procedure with 9) ENTER to return to normal mode.

DISPLAY ON SCREEN KEYS USED

1)



2) GRAPH
3)
4) AND
5)

6)
7)
8) enter new max. address with HEX PAD
9)

Selective data acquisition

With selective data acquisition, you can use the memory in a very efficient way since you can make it store only the data you are interested in.

Do the following:

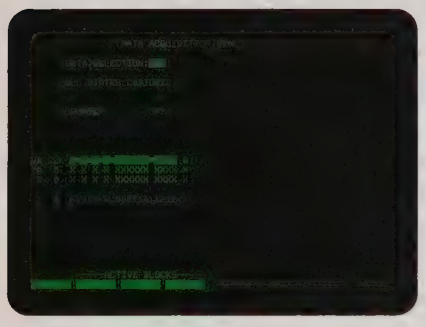
1) Press DATA
2) (if the whole data menu is not displayed on screen press FULL)
3) Press ACTIVE BLOCKS 1
4) Move cursor to "clocks"
5) Press ONLY WORDS
6) Place cursor on WORD A
7) Enter the kind of DATA that you want to observe
8) Press SINGLE
9) Press LIST and watch the result

In the Philips target, for example, you enter 0 63X5 XXXXXXXX XXXX X F in word A, which will offer you all instructions in analyzers memory without datatransfers involved.

You can stop the selective data acquisition by:

10) Press the ⌨ -key
11) move cursor to 'block 1'
    and
12) OFF

DISPLAY ON SCREEN | KEYS USED



1) DATA
2) FULL

3)
4) Move ◂▸ cursor to "clocks"
5)
6) Place ◂▸ cursor on word A
7) Enter with HEXPAD and selective data

8) SINGLE
9) LIST

and finish with:

10)
11) Move ◂▸ to "block 1"
12)

Comparison of state data

The compare facility for state analysis is a powerful tool for the location of system faults. It works by comparing two different state shots, one stored in the acquisition memory, the other in the reference memory.

The first thing that must be done is to save one of the acquisitions in the reference memory:

- Press SINGLE key to display a new acquisition and press COPY TO REF to save it.

Then proceed with:

1. COMPARE
2) FULL
3) Move cursor to 'state section'
4) Press ON

Now the comparison in the state analysis mode has been activated.

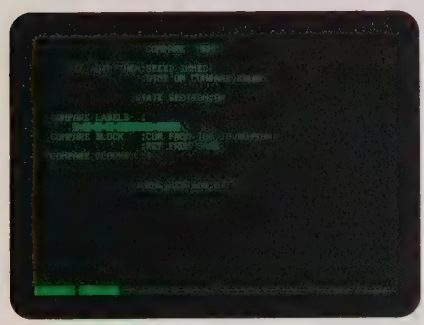
If you want to compare all the data, then leave the compare labels unchanged. Continue with:

5) Move cursor to 'TOP'. Then press LINE.
6) Move cursor to 'compare clocks' and change 0 in 0+1
7) SINGLE, which means: do an acquisition
8) LIST
9) COMPARE

You can observe that the comparison is equal (top of the screen on the right).

Until now you have just compared two equal program flows. Now change the captured data somewhat and make another acquisition with SINGLE. On screen the differences between both of them can be seen clearly, since they are highlighted.

DISPLAY ON SCREEN KEYS USED



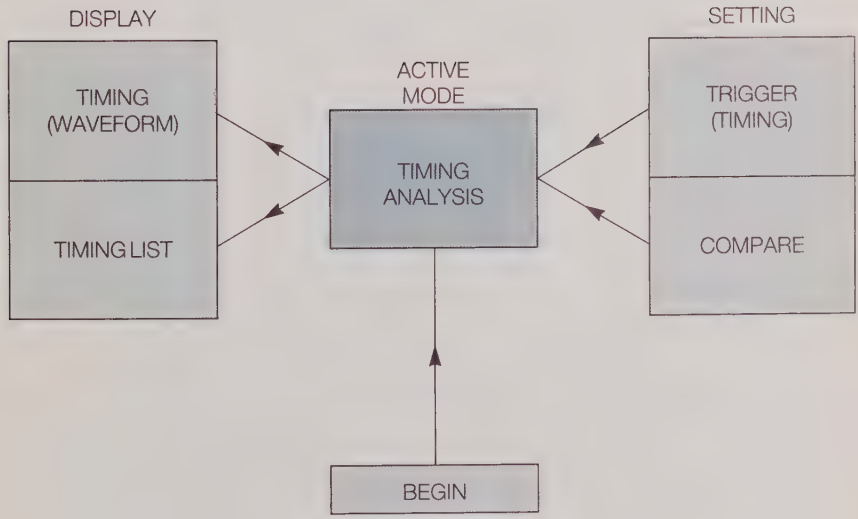
■ SINGLE and COPY TO REF
1) COMPARE
2) FULL
3) Move ◁▷ cursor to 'state SECTION'
4) [illegible]

5) Move ◁▷ cursor to 'TOP' and press [illegible]
6) Move ◁▷ cursor to 'compare clocks' and press [illegible]

7) SINGLE
8) LIST
9) COMPARE
Now change captured data
10) SINGLE

TIMING ANALYSIS

1) Introduction
2) Making and displaying an acquisition
3) Timing triggering
4) Comparison of timing data



Introduction

In timing analysis, data is sampled from the microprocessor under test under control of an internal clock. This clock can sample at a maximum sample rate of 400 MHz, which means you are able to measure with a 2.5 ns resolution.

Also an external generator can be connected to the PM 3570 to perform timing measurements for applications that require very low clock speeds (from DC to 100 MHz).
(For further information see Operating Manual part 5).

As well as this, the PM 3570 has transitional timing analysis, which provides you with an enormous memory depth. Transitional timing is based on the fact if nothing changes in the received data, only the time during which nothing has changed, needs to be stored.

Making and displaying an acquisition

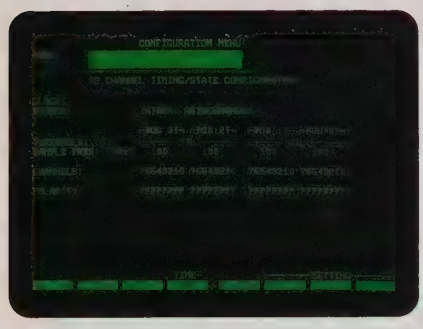
Once again it is assumed that you proceed after the start-up, followed by:

1) Press 32 CHAN to enter the timing section. Note that all the channels are installed at an internal asynchronous clockrate of 100 MHz.
2) Press SINGLE to make an acquisition
3) Press TIMING to receive a timing diagram
4) Use BEST to optimise your display
   On screen 16 channels are displayed. In total the analyzer receives 32 timing channels. They are all presented to you in the timing table, which can be reached by:
5) press SEQ
6) press TABLE

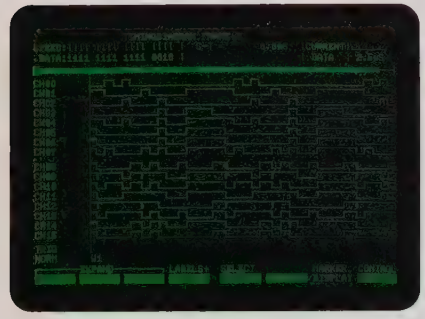
Four default frames are given, but you can create your own frame simply by changing the channel numbers in a frame. You can also rename the channels in the frames.

We wish to use only 8 channels, therefore we use:
7) SELECT FRAME
8) FRAME 3

DISPLAY ON SCREEN KEYS USED



1) [redacted]



2) SINGLE
3) TIMING
4) [redacted]

5) [redacted]
6) TABLE
and select with:
7) [redacted]
8) [redacted]

Timing triggering

As in state analysis, it is possible to trigger at any required moment. This can be done in the Timing Triggering Menu.

To display the Trigger Menu on screen, first exit from the Table Display by

1) TIMING DISPLAY, then press
2) TRIGGER and
3) FULL and a number of possible trigger sequences are shown. We will use predefined trigger sequence A, but you can choose your own.
   Entering a trigger word can easily be done by:
   4) Press the TIMING key and (if not already done) select frame 3
   5) Move cursor to the place you are interested in
   6) When trigger place is found press ☰ -key
   7) Press COPY TO TRIGW to save the required trigger word

If, for example, you want to see other timing channels, you should first press ☰ followed by SELECT FRAME. By pressing one of the softkeys, another frame of channels will be displayed on screen. To return to normal mode press RETURN.

To return to your trigger 'moment' do the following:

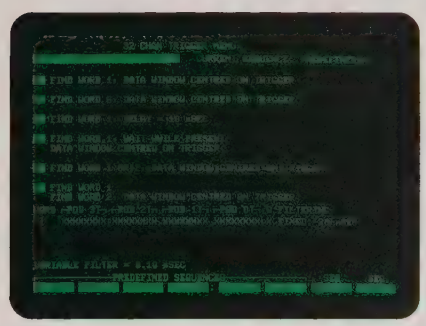
8) Press ☰ and CURSOR CONTROL
9) Press PUT CURSOR
10) Press ON TRIGGER

and your cursor will move directly to the trigger word.

Frame 0 provides you here with the lower 16 measured timing channels, and frame 1 with the higher 16. When the pods are installed as described in Hook-up with target, frame 0 shows you the data lines of the M68000, and frame 1 the first 16 address lines.

As already stated you can use the timing pods to measure other signals simply by changing the pods at the rear of the analyzer and on the personality adaptor of the micro under test (see Hook-up with the target).

DISPLAY ON SCREEN KEYS USED



1)
2) TRIGGER
3)

4) TIMING
5) Move ◂ ▸ cursor to new triggerword
6)
7) COPY TO TRIGW

8) Press ⏎ and
9)
10)

Comparison of timing data

You can easily compare two different acquisitions in timing mode, using one acquisition in your reference memory and the other in your current memory.

Perform the following steps:

1) Press COMPARE
   The complete compare menu is displayed on the screen.
   (If not, then press also the FULL key)
2) Move down with your cursor to the timing section and switch it on by ON
   Some variables are shown in the timing section of this menu. You can use their default values, but you can also:
   a) compare two acquisitions beginning a time interval after the trigger word (from TRIG')
   b) compare during another time interval (LENGTH)
      and
   c) compare with greater tolerance (SKEW)
   It is also possible to compare not all the channels, but just a few by switching off the ones that are not desired.

■ Move cursor to 'LENGTH' and change nano sec into microsec using μSEC

3) Press TIMING and CONTROL DISPLAY
4) Get frame 3 on screen: SELECT FRAME and FRAME 3
5) Press SINGLE to receive new data while using current trigger word
   Now save your current data in the reference memory by:
6) COPY TO REF

To see the effect of comparison change the captured data and do another acquisition with

7) SINGLE

Finally:

8) Press COMPARE and watch the changes

The differences between both acquisitions are blinking on your screen. Moreover the channels that are not equal are also blinking.

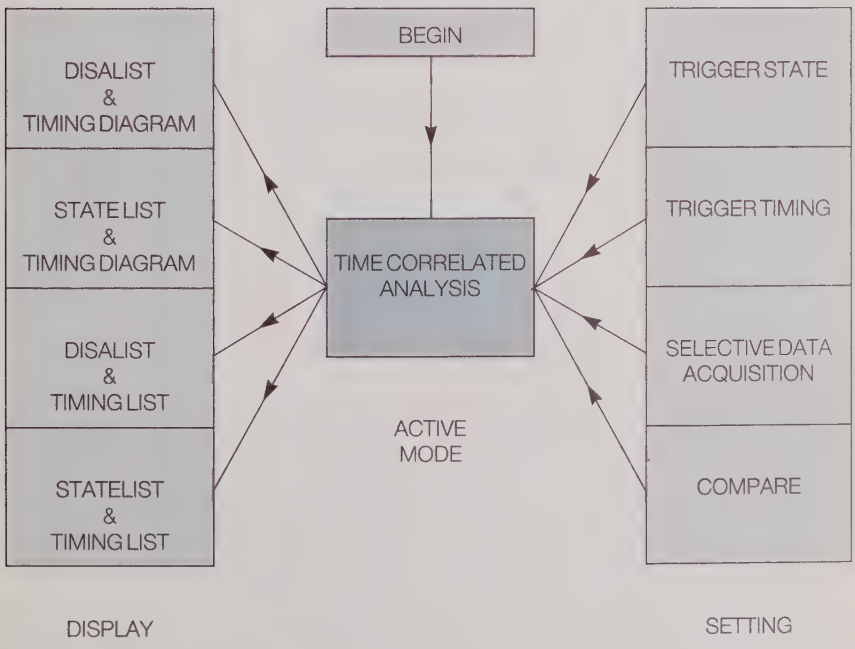
DISPLAY ON SCREEN KEYS USED

1) COMPARE
2) Move ◁▷ cursor to timing section and press
- Now change 1000 n SEC into 1000 μ SEC using ◁▷ and

3) TIMING and
4) and
5) SINGLE
6) COPY TO REF

Now change captured data
7) SINGLE
8) COMPARE

1) Introduction
2) Making and displaying an acquisition



Introduction

In time correlated mode state and timing measurements are correlated. This means that you are able to analyse them simultaneously.

The starting point, once again, is just after the start-up instructions mentioned in section I. Now press TIME CORREL. and look at the configuration menu. Proceed by:

1) Press TRIGGER. You are in the state trigger menu and here you must enable the timing triggering by:
2) Place cursor on 'DELAY'. Press ENABLE and DELAY.

Now modify the timing triggering by pressing:

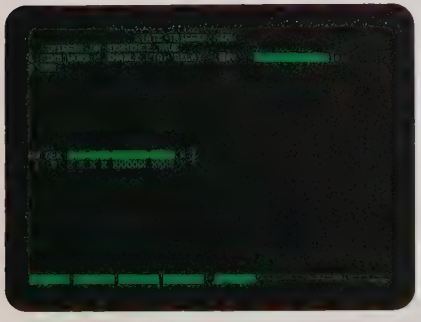
3) TRIGGER again (notice the toggle function again: trigger menu in two parts, both reached by TRIGGER)
4) SEQ DISPLAY
5) Move cursor to 'FIND' and press LSA WAIT

Now you are ready for time correlated acquistion.

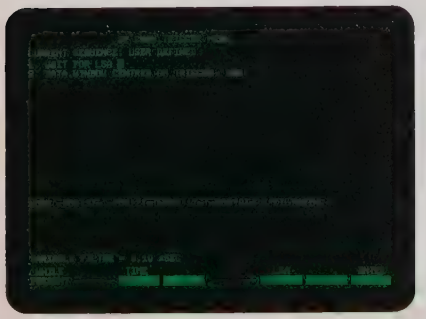
Note: This analysis mode is ideal to measure hardware- and software-related faults, such as capturing a glitch and analysing its influence on the program flow or capturing wrong state data and checking of the timing faults, which are responsable.

DISPLAY ON SCREEN KEYS USED

■



1) TRIGGER
2) place cursor on 'DELAY' and press followed by



3) TRIGGER
4)
5) move cursor to 'FIND' and press

Making and displaying an acquisition

After setting up both the state and the timing trigger menu, you can make an acquisition by pressing 1) the SINGLE key.
Displaying the results in time correlated mode can take place in several ways.
The first of these is done as follows:

2) Press TIMING key, and a timing diagram is displayed on screen
3) Press SELECT FRAME
4) Press FRAME 3
5) press BEST

Since you only display eight channels in a timing diagram, there is room available at the top of the screen to insert a state list:

6) Press the -key
7) Press LIST and the time stamped state list is displayed at the top of the screen

At the right side of the state list, you can see the time at which the timing analysis triggered. In this case, triggering was between the state lines 2 and 3.

The time correlated mode operates with two separate displays, and you can modify these with the softkeys. Those used to change the display of the state list are reached by pressing the -key, and those used to change the timing diagram are reached by pressing the -key.

This possibility allows the displays to be changed seperately, so that different display methods can be combined. The first of these is now on screen, but you may wish to display a disassembly list, so press:
8) The -key (to reach the softkeys in the state field)
9) DISA

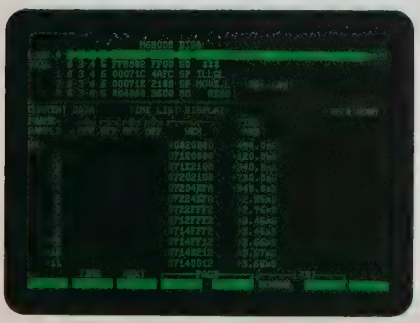
Now the disa list replaces the time-stamped state list.
Moreover you can alter the timing diagram in a timing list by pressing:

10) The -key (to reach the softkeys in the timing field)
11) LIST

DISPLAY ON SCREEN KEYS USED

1) SINGLE
2) TIMING
3)
4)
5)
6)
7) LIST

8)
9)



10)
11)

Another very important feature of the time correlated mode is the fact that you can scroll state and timing data fields. Once again the double function of the softkeys is mentioned, reached by the ☰ -key and the ☰ -key.

1) Press ☰ -key
2) Press TIMING
3) Press MARKER DISPLAY
4) Press MARKERS SYNC

The timing display and the state list are synchronised, so when the timing display is scrolled, the state list follows and vice versa.

Of course you must also be able to display other channels than those which frame 3 offers as default. This can easily be done by pressing:

5) LABELS SEQ

Now the channels you are observing at present are shown, just above the timing display. Place the cursor on the channel you want to change and enter the channel number you want to display, by using the appropriate hex key (0 to F). You can leave this mode to normal mode by pressing 6) RETURN.

DISPLAY ON SCREEN

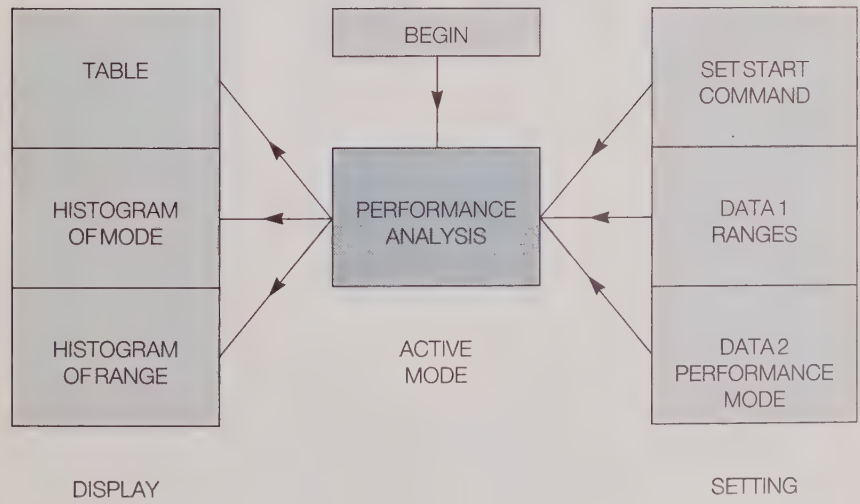
KEYS USED

1) [redacted]
2) [redacted]
3) [redacted]
4) [redacted]

Now scroll with ◁ ▷ through timing

5) [redacted]
Change with HEXPAD current
Channel in desired channel
6) [redacted]

1) Introduction
2) Event frequency analysis
3) Execution period analysis
4) Displaying results in a histogram



Introduction

The third logic analysis mode, performance analysis (P.A.), allows you to optimise your software code. Starting in the configuration menu, press 1) PERF. AN. to enter P.A.

Then, proceed by pressing the DATA key. The data menu is the most important menu available in P.A.. In fact two different menus can be displayed alternately; the first one to define the ranges, the second to define the required P.A. mode.

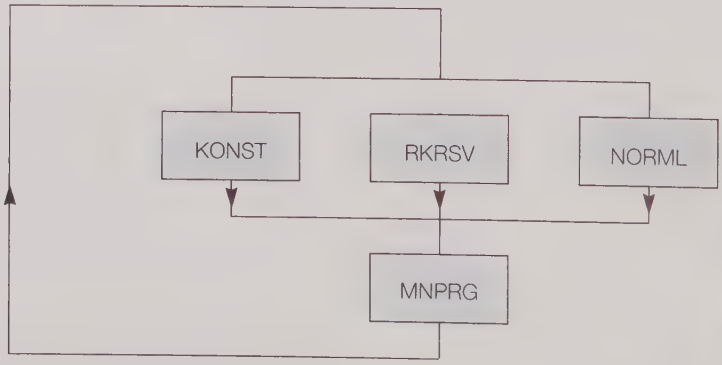
A. This is the menu displayed on screen by pressing the 2) DATA key once. Enter your ranges here. A range is a part of the program in whose performance you are interested.
It is defined by a begin and an end address.
To insert these in the menu, move the cursor to '>use' and then press +. A new range appears on screen. Then define its name and the begin and end addresses by using the softkeys.

In our example we use the ranges:

	start	end
KONST	0008A6	0008C8
RKRSV	0008CC	0008EE
NORML	0008F2	000914
MNPRG	00088E	0008A2

Using the M68000 target system, first press the reset button, and then the INT4 button to start a special P.A. demo program. Of course, you can enter any range you wish to test your own program.

Our example program flow:



DISPLAY ON SCREEN KEYS USED

1) [redacted]

2) DATA

3) Move ◁▷ cursor to '>USE'
4) [redacted]
Now use HEXPAD and SOFTKEYS to enter ranges

Event-frequency analysis

2) The second P.A. menu can be displayed by pressing 1) the DATA key again.

At the bottom of the screen, the available options are shown. The default option is event frequency analysis. This means the analyzer counts, during a specific time interval, the number of events from each specified range. Both the time interval and the required ranges can be adjusted in this menu.

Continue now with an acquisition 2) (SINGLE) and display of the results by pressing the 3) LIST key.

Since P.A. measurements are statistical measurements, it is sometimes necessary to acquire more data. So press 4) AUTO and wait until the values are more or less stable. Then the results will be reliable.

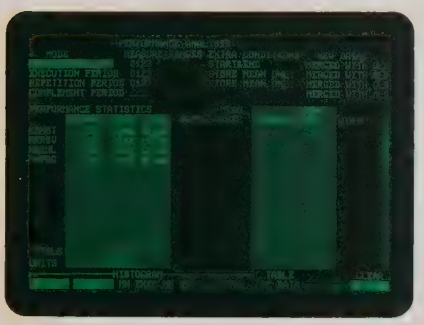
Event frequency:

6 events during specific time interval

DISPLAY ON SCREEN KEYS USED

1) DATA

2) SINGLE
3) LIST



4) AUTO

Execution period analysis

As well as measuring the event frequency, it is possible to use another P.A. mode of the analyzer: execution period analysis.

This method determines the main and individual values of the execution period for each defined range.

Actions:

- Shift back to the DATA menu by pressing 1) FULL. Then change 'event frequency' to 'execution period' by 2) moving the cursor to 'event' and 3) pressing EXEC.
- Also change 'storing mean value only' into 'each value', using the softkey ALL. Now, once again, push 6) SINGLE and 7) LIST to display the required results on screen.
- Again, for statistical reasons, you should acquire several results by pressing 8) AUTO. Together with the value found for the event frequency, the value of the execution period gives you the total value which is on the screen under the column TOTAL.

In our case, we have chosen a time interval of 1 second, and the total value appears to be 908 msec. So our test program consumes nearly 91% of the available system cycle time, while 9% is available for system code extension, without slowing down overall system performance.

Execution period:

mean execution time: $\frac{T1+T2+T3+T4+T5+T6}{6}$

Note: Performing execution period analysis in the 'storing mean value only' mode offers you upto 65535 samples, while in 'store all' mode this is 99 samples maximum.

DISPLAY ON SCREEN KEYS USED

1) [redacted]
2) Move cursor to 'Event'
3) [redacted]
4) Move ◁▷ cursor to 'STORING MEAN VALUE ONLY'
5) [redacted]

6) SINGLE
7) LIST

8) AUTO

Displaying results in a histogram

It is also possible to display a histogram of the measured data on screen.
To do this push EVENTS.

- To get a useful display: push EXPAND, BEST and READY.

You might also wish to use these steps for the main execution period.

- Then use: MN EXEC, EXPAND, BEST, READY.

Finally you can observe all the results of one particular P.A. mode. This means you can get the statistical distribution of the measured values from all the defined ranges separately.

For example: You are interested in the distribution of the results in the measurement of the MNPRG execution period.

This can easily be done by pressing:

1) GRAPH key
2) EXEC
3) now select the kind of range for which you would like a diagram to be displayed.

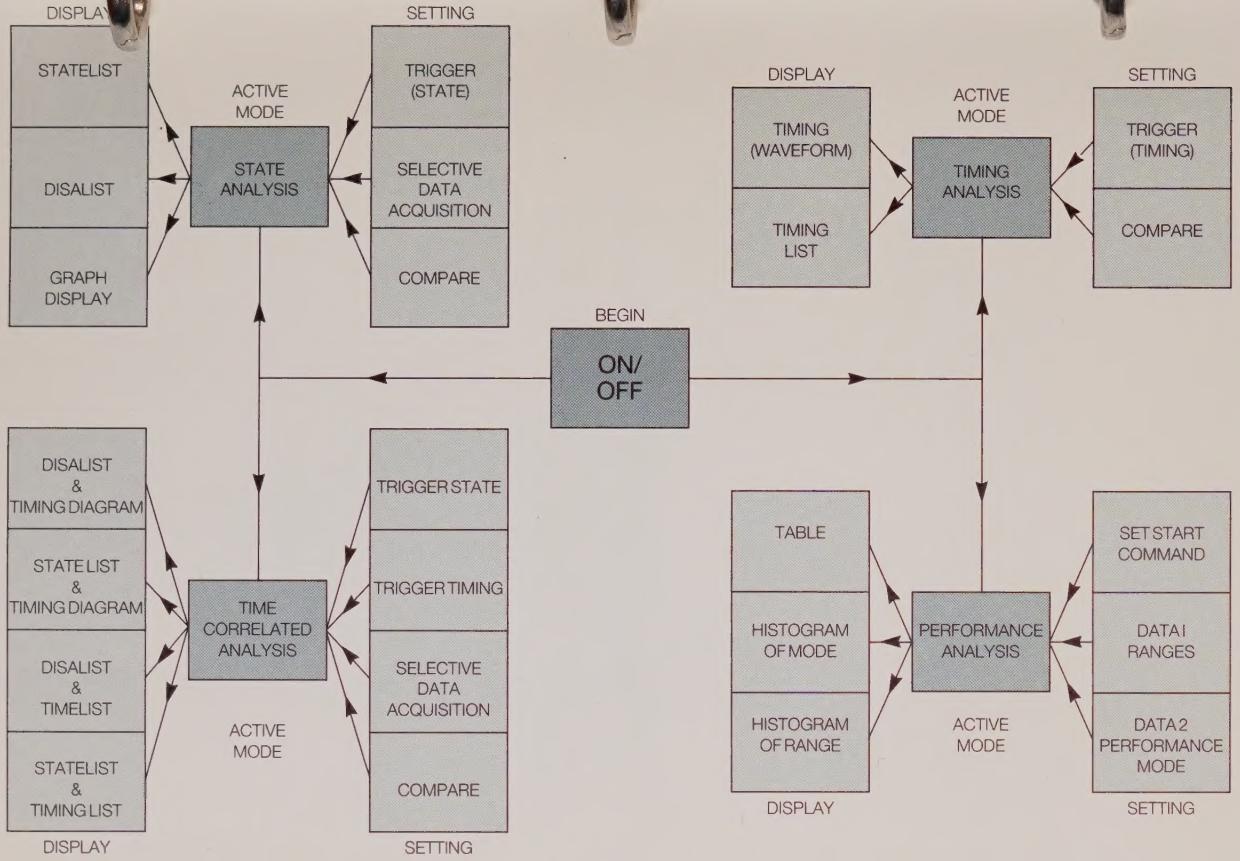
Now all relevant information about the specific range appears on your screen. By using the values of mean, min and max in particular you can modify the vertical interval to make an optimal displayed histogram for this range.

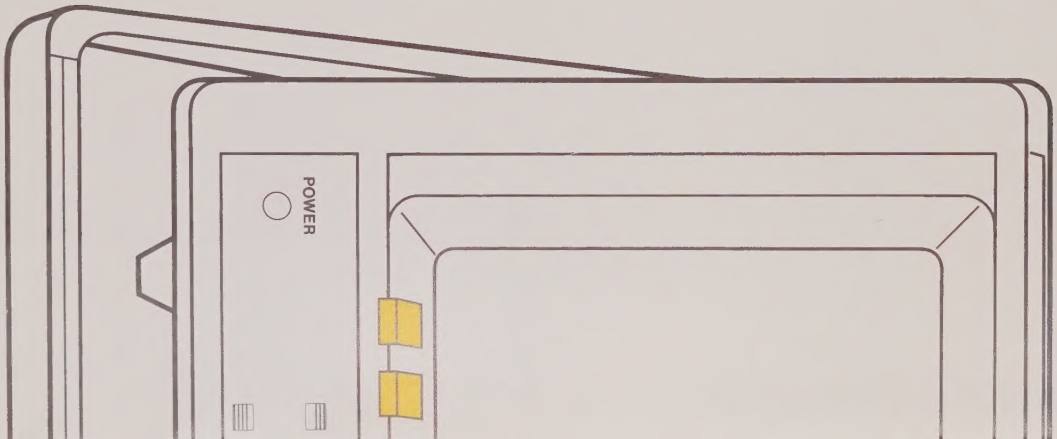
4) MODIFY INTERVLS
5) enter new top value
6) choose the necessary dimension by using a softkey
7) BOTTOM
8) CLEAR
9) enter a well-fitted minimum value with its dimension
10) TOP, you should return to your top value
11) EQUAL INTRVLS
12) READY, press ready to return to normal mode

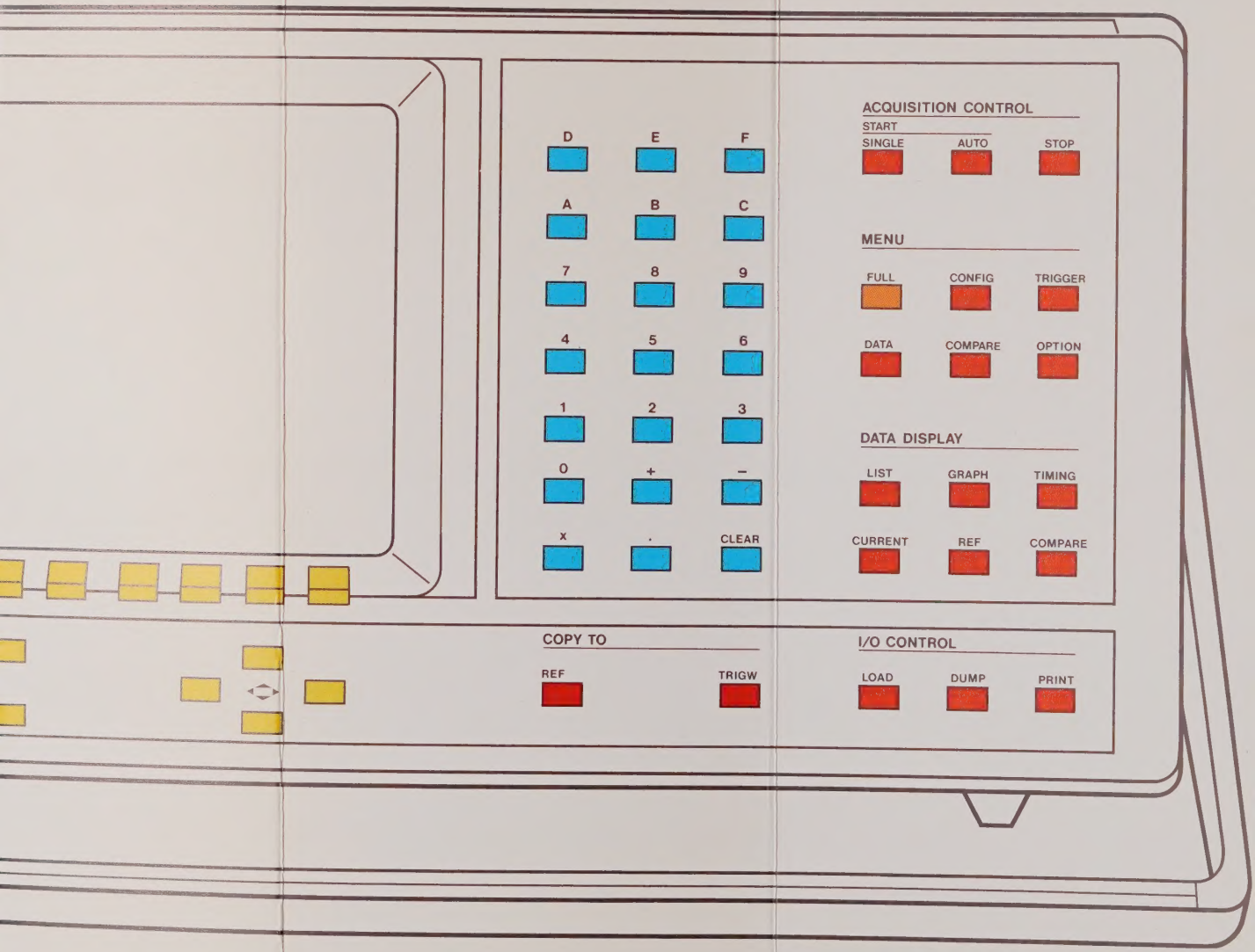
Changing the horizontal scale can easily be done by using the softkeys MORE, LESS or BEST.

DISPLAY ON SCREEN KEYS USED

1) GRAPH
2)
3) Select RANGE with HEXPAD
4)
5) ENTER TOP VALUE HEXPAD
6) Softkey with dimension
7)
8) CLEAR
9) ENTER minimum with HEXPAD and its dimension
10)
11)
12)


DISPLAY
STATELIST
DISALIST
GRAPH DISPLAY
ACTIVE MODE
STATE ANALYSIS
SETTING
TRIGGER (STATE)
SELECTIVE DATA ACQUISITION
COMPARE
DISPLAY
TIMING (WAVEFORM)
TIMING LIST
ACTIVE MODE
TIMING ANALYSIS
SETTING
TRIGGER (TIMING)
COMPARE
BEGIN
ON/ OFF
DISALIST & TIMING DIAGRAM
STATE LIST & TIMING DIAGRAM
DISALIST & TIMELIST
STATELIST & TIMING LIST
DISPLAY
TIME CORRELATED ANALYSIS
ACTIVE MODE
TRIGGER STATE
TRIGGER TIMING
SELECTIVE DATA ACQUISITION
COMPARE
SETTING
TABLE
HISTOGRAM OF MODE
HISTOGRAM OF RANGE
DISPLAY
PERFORMANCE ANALYSIS
ACTIVE MODE
SET START COMMAND
DATA I RANGES
DATA 2 PERFORMANCE MODE
SETTING


POWER


D
E
F
A
B
C
7
8
9
4
5
6
1
2
3
0
+
−
x
.
CLEAR
ACQUISITION CONTROL
START
SINGLE
AUTO
STOP
MENU
FULL
CONFIG
TRIGGER
DATA
COMPARE
OPTION
DATA DISPLAY
LIST
GRAPH
TIMING
CURRENT
REF
COMPARE
COPY TO
REF
TRIGW
I/O CONTROL
LOAD
DUMP
PRINT

MP MASTER
MA24616